PEL VISFLO

User Guide

About this document

This guide is designed to assist the user in becoming quickly familiar with the capabilities of VisFlo, its interface and how the program is used.

It has been produced to the recommendations of British Standard BS7649 – Guide to the design and preparation of documentation for users of application software.

Trademarks

All trademarks acknowledged.

.

Contacting PEL Support Services

This program is developed, maintained and supported by PEL Support Services, ABB. We run a Hotline telephone and email service to answer any queries about VisFlo.

Please let us have any suggestions on how you feel we could improve VisFlo. You can contact us by any of the following routes:

By Telephone: ++44 (0)1925 74 1126

By Fax: ++44 (0)1925 74 1265

By E-mail: pel.support@gb.abb.com

By Post: PEL Support Services
ABB Ltd.
Daresbury Park
Daresbury
Warrington
Cheshire
WA4 4BT
United Kingdom.

Owner:	M. G. Pass, ABB
Approved By:	M. G. Pass, ABB
Document Version / Issue Date:	Document Version 1.4 / 24 October 2005
Last Amended Date:	24 October 2005
Last Amended By:	G. Langford, ABB

Change history

This table records the changes made to each new revision of this document.

Changes to approved issues are indicated by a double revision bar on the outer margin next to the text. This is an example.

Revision	Date	Description of change
1.0	07 March 2000	First Approved Issue
1.1	08 March 2001	Second Approved Issue. This issue completely replaces previous versions of this document as the user interface has been significantly upgraded.
1.2	21 March 2001	Third Approved Issue (ABB logo added).
1.3	10 October 2002	Fourth Approved Issue (Industrial IT logo & paragraph added, "Eutech" removed, Front page modified.)
1.4	24 October 2005	Removed Industrial IT logo and paragraph.

Contents

Figures

Appendices

1. VisFlo User Guide

1.1. Introduction

The VisFlo program can be used to perform pressure drop and heat transfer calculations on Newtonian and non-Newtonian liquids in pipe flow.

The program has three modes of operation:

Isothermal Pressure Drop	VisFlo can be used to calculate pressure drop from flowrate or vice versa for isothermal flow in a pipe system. Both laminar and turbulent flow can be handled and a variety of pipe fittings are available.
Laminar Flow Heat Transfer	VisFlo can be used to calculate pressure drop and heat transfer for laminar flow in straight pipes. Either wall temperature or wall heat flux boundary conditions can be specified.
Turbulent Flow Heat Transfer	VisFlo can be used to calculate pressure drop and heat transfer for turbulent flow in straight pipes.

Details of the calculation procedures for the three modes of operation are given in the relevant chapters of the non-Newtonian flow design guide. The fluid viscous properties are assumed to be described by the Generalised Bingham model:

$$\tau = \tau_Y + K * \gamma^n$$

where:

τ	–	Shear Stress
τ_Y	–	Yield Stress
K	–	Consistency Index
γ	–	Shear Rate
n	–	Flow Behaviour Index
If $\tau_Y = 0$	–	the model represents the well know power-law behaviour.
If $\tau_Y \neq 0$ and $n = 1$	–	the model represents Bingham Plastic behaviour.
If $\tau_Y = 0$ and $n = 1$	–	the model represents a Newtonian fluid and K is the absolute viscosity.

In the laminar flow heat transfer calculation the consistency index is considered to be a function of temperature and defined by the equation:

$$K = K_0 * e^{[n*A*(\frac{1}{T} - \frac{1}{T_0})]}$$

where:

K_0 = a reference consistency index at temperature T_0
A = the temperature dependence parameter (K)
T_0 = the reference temperature (K)
T = the fluid temperature (K).

Both τ_Y and n are assumed to be independent of temperature.

In the turbulent flow heat transfer calculation, the viscous properties are assumed to be independent of temperature. Fluid properties at the average bulk temperature should be used.

In all three modes of operation the user has the option of supplying either the generalised Bingham model parameters described above or absolute viscosities for a Newtonian fluid.

VisFlo uses SI units for input and output except that ***all*** temperatures including the reference temperature are in Degrees Centigrade.

1.2. How this guide is structured

This guide is designed to assist the user in becoming quickly familiar with the capabilities of, its interface and how the program is used.

The chapters are organised as follows:

Chapter 1	An introduction to VisFlo.
Chapter 2	Details the VisFlo user interface.
Chapter 3	A tutorial to guide the user through a typical VisFlo session emphasising the commonly used features. It is recommended that the user should read this chapter while running the program.

2. The VisFlo User Interface

The VisFlo user interface displays tabs that allow easy definition of the physical properties being calculated. The following sections of this chapter contains detailed information on the functions available with the dialogues assigned to these tabs.

On-line documentation is available within the program.

2.1. The VisFlo Start up Screen

The VisFlo start up screen (see Figure 1) consists of the following:

Menu Bar	Displays the available options.
Toolbar Buttons	Descriptive text appears automatically as the cursor is held over the buttons describing their function.
Tree View Display	Used to store and retrieve calculations generated by the user or brought in from file. Click on the plus sign to expand the tree, the minus sign to collapse it and click on the calculation in the list to display it.
Calculation Data Input Tabs	Clicking on these tabs displays the input screens for the type of calculation selected.
Display Calculation Results	Displays the calculation results.
Status Bar	Displays program status information.

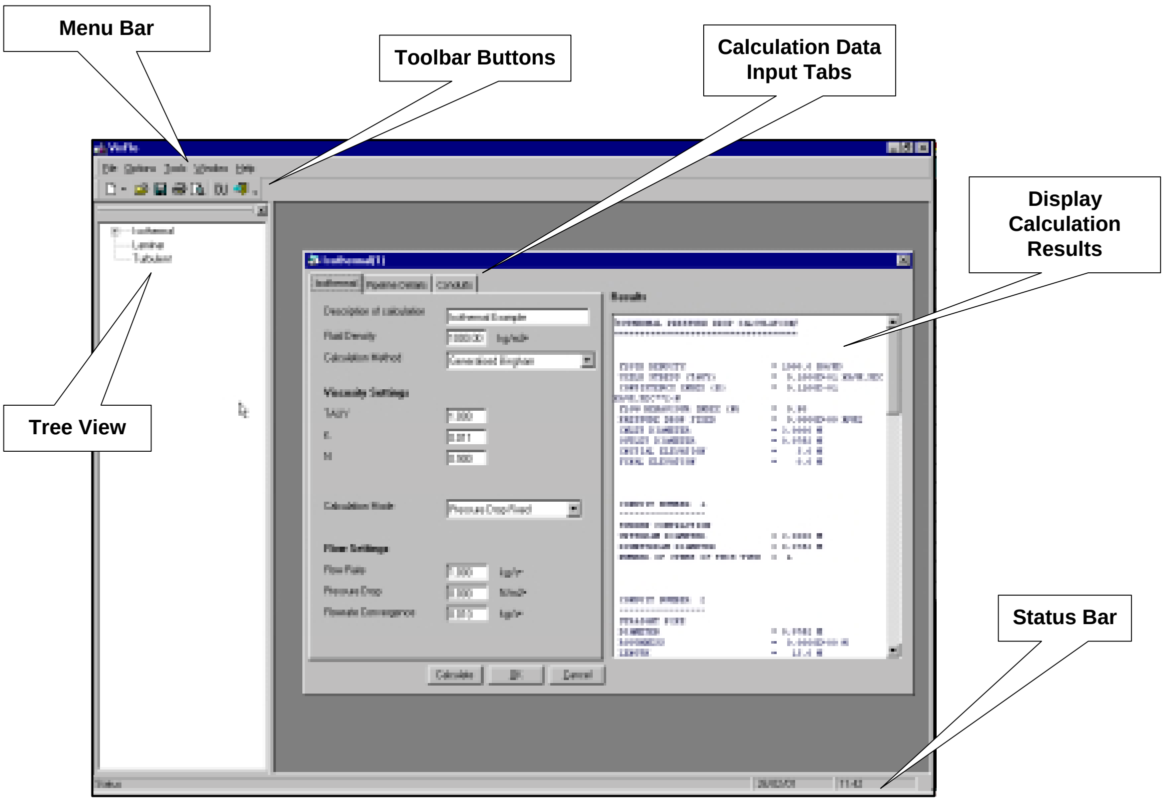


Figure 1 VisFlo Start up screen

The following sections of this chapter contain detailed information on the functions available within these tabs.

2.2. Isothermal Calculation

Used to calculate pressure drop from flowrate or vice versa for isothermal flow in a pipe system.

Click on the plus sign to expand the tree, the minus sign to collapse it and click on the calculation in the list to display it.

Note. Clicking on a calculation from the tree view more than once will duplicate an instance of it and make it available for comparing results with slight modifications to input.

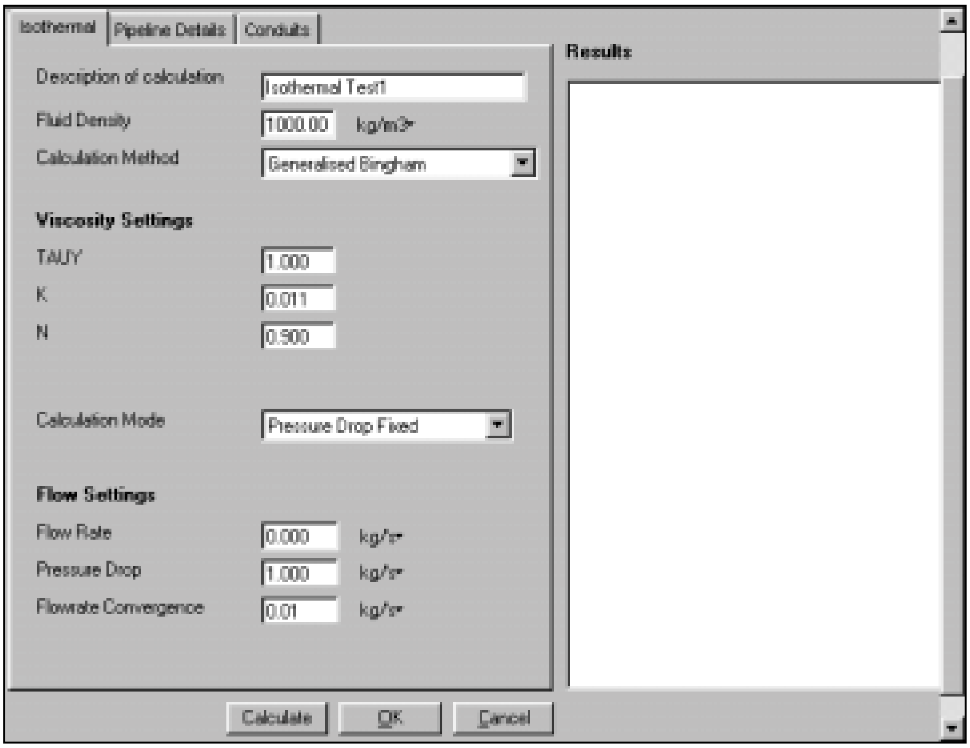


Figure 2 Isothermal Calculation dialogue

The following selections can be made from this dialogue:

Usage

Description of calculation	Mandatory Enter a text string to identify the calculation.
Fluid Density	Mandatory. The actual density of the fluid at operating temperatures. The units can be chosen as appropriate.
Calculation Method	Mandatory. Select the Calculation Method from either **Newtonian Fluid** or **Generalised Bingham**. The following selections are available when **Newtonian Fluid** is selected: **Viscosity Settings** • Viscosity at Bulk Temperature. **Calculation Mode** Select either **Flow Rate Fixed** or **Pressure Drop Fixed** from the drop down menu. • Flow Rate Fixed Enter values under **Flow Settings** for: - Flow Rate. • Pressure Drop Fixed Enter values under **Flow Settings** for: - Flow Rate - Pressure Drop - Flowrate Convergence. The following selections are available when **Generalised Bingham** is selected: **Viscosity Settings** • TAUY • K • N **Flow Settings** • Flow Rate • Pressure Drop • Flowrate Convergence.
Calculate	Performs the calculation and displays any results in the Results field.
OK	Saves the calculation and the entered data to the tree view.
Results	The field where the calculated results are displayed (see Figure 13 for an example).

2.2.1. Isothermal Calculation – Pipeline Details Tab

Used to add Pipeline details for the Laminar calculation.

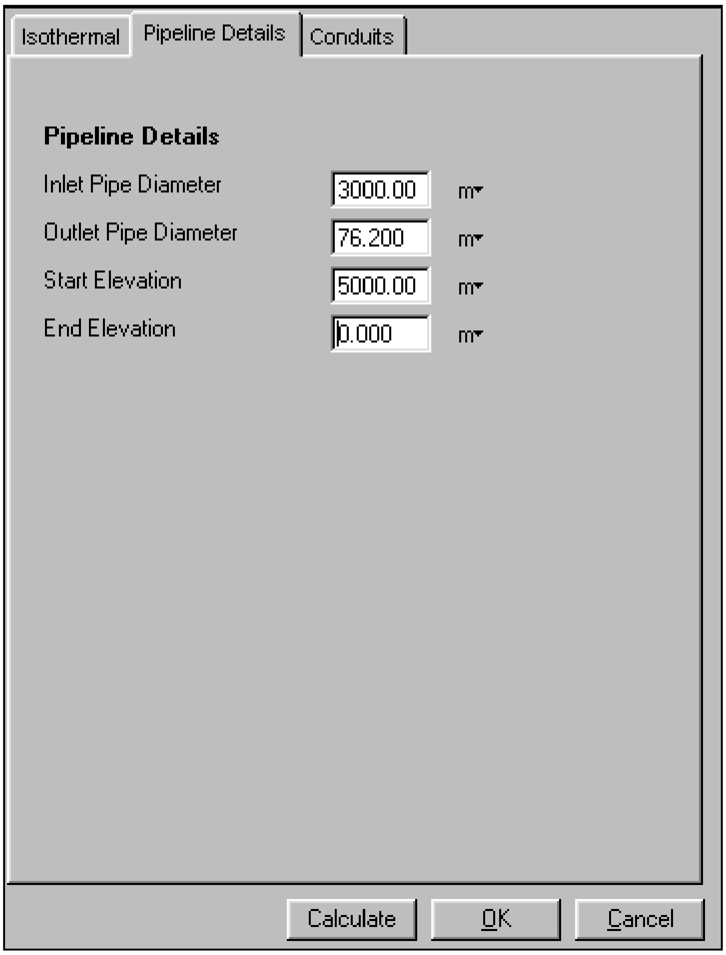


Figure 3 Pipe Details dialogue

The following selections can be made from this dialogue:

Usage

Pipeline Details	Mandatory. Enter values for: • Inlet Pipe Diameter The internal pipe diameter of the inlet pipe. • Outlet Pipe Diameter The internal pipe diameter of the inlet pipe. • Start Elevation The height of the inlet pipe above ground level. • End Elevation The height of the outlet pipe above ground level.
Calculate	Performs the calculation and displays any results in the Results field.
OK	Saves the calculation and the entered data to the tree view.

2.2.2. Isothermal Calculation – Conduits Tab

Through the display, conduit types and data can be entered, removed and sorted in the order the pipeline system and components are assembled.

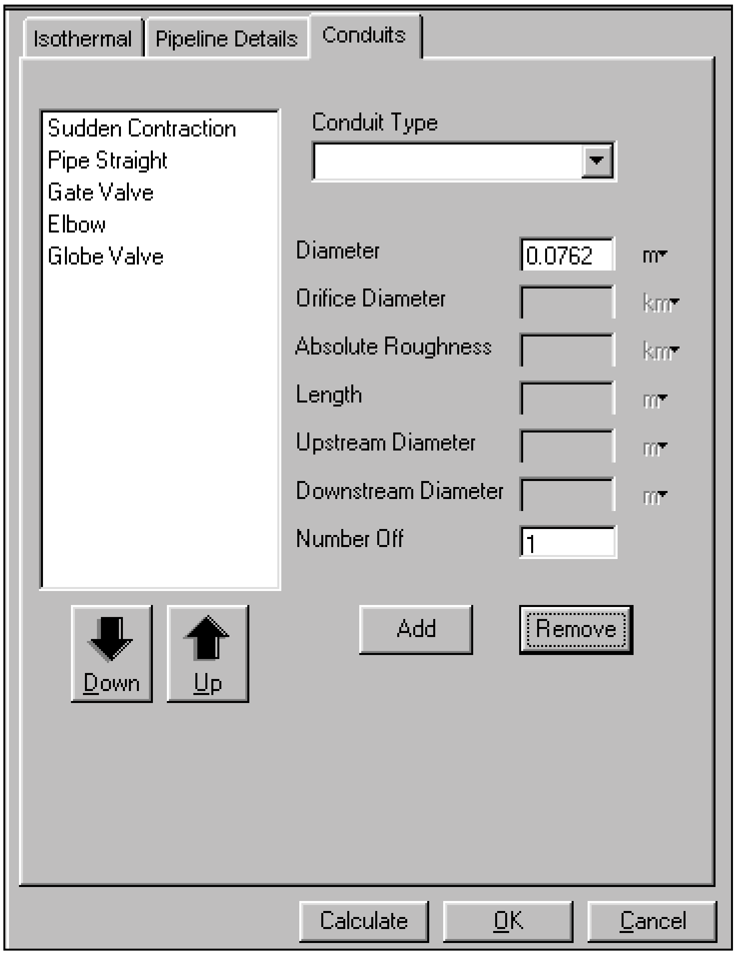


Figure 4 Conduits dialogue

The following selections can be made from this dialogue:

Usage

Conduit Type	Mandatory. The following **Conduit Types** can be selected: • Pipe Straight • Sudden Contraction • Sudden Expansion • Orifice Plate • Elbow • Gate Valve • Globe Valve Enter values for: • Diameter • Orifice Diameter • Absolute Diameter • Length • Upstream Diameter • Downstream Diameter • Number Off. Certain boxes may be greyed out depending on the Conduit Type selected.
Add	This button is used to Add a conduit to the list. • Select one of the seven conduit types from the pull down list (for example, Pipe Straight). • Enter data appropriate to the conduit selected. • Click on the **Add** button to add the conduit. **Note.** Ensure that all fields have data entered or the conduit will not be added to the list.
Remove	This button is used to Remove a conduit from the list. Select the conduit from list on left-hand then click on the **Remove** button to remove from list.
Up / Down	This button is used to reorder the Conduit list. Select the conduit from the list on left-hand side then use the Up 🡅 and Down 🡇 buttons to reposition it in the list.

2.3. Laminar Calculation

Used to calculate pressure drop and heat transfer for laminar flow in straight pipes. Either wall temperature or wall heat flux boundary conditions can be specified.

Click on the plus sign to expand the tree, the minus sign to collapse it and click on the calculation in the list to display it.

Note. Clicking on a calculation from the tree view more than once will duplicate an instance of it and make it available for comparing results with slight modifications to input.

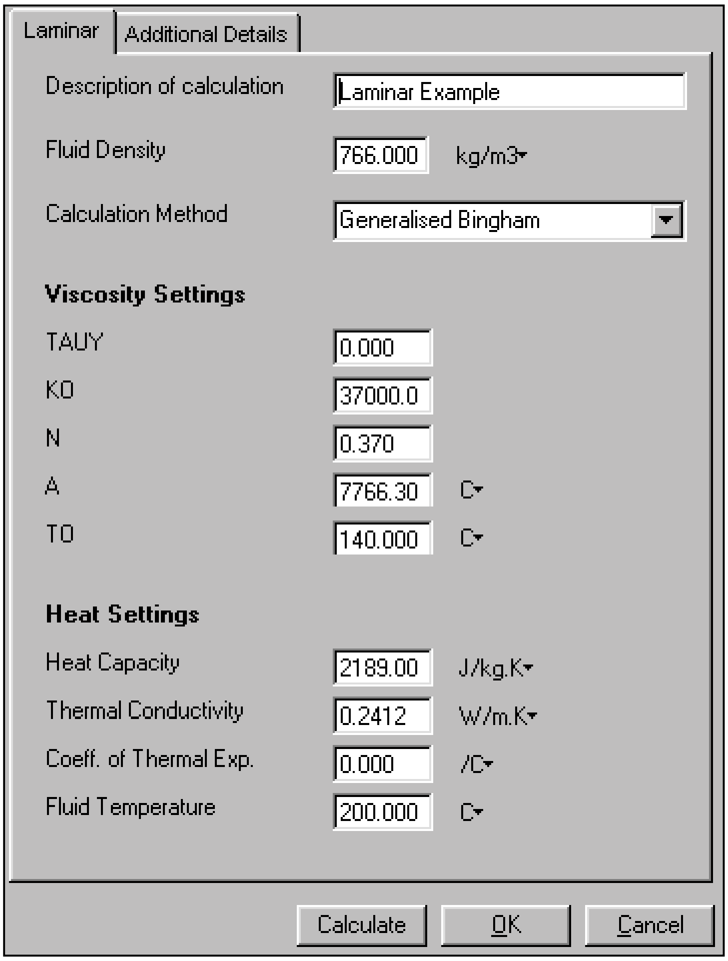


Figure 5 Laminar Calculation dialogue

The following selections can be made from this dialogue:

Usage

Description of calculation	Mandatory. Enter a text string to identify the calculation.
Fluid Density	Mandatory. The actual density of the fluid at operating temperatures.
Calculation Method	Mandatory. Select from either **Newtonian Fluid** or **Generalised Bingham**. The following selections are available when **Newtonian Fluid** is selected: **Viscosity Settings** • 1^{st} and 2^{nd} Viscosity Values • 1^{st} and 2^{nd} Temperature Points. **Heat Settings** • Heat Capacity • Thermal Conductivity • Coefficient of Thermal Expansion • Fluid Temperature. The following selections are available when **Generalised Bingham** is selected: **Viscosity Settings** • TAUY • KO • N • A • TO. **Heat Settings** • Heat Capacity • Thermal Conductivity • Coefficient of Thermal Expansion • Fluid Temperature.
Calculate	Performs the calculation and displays any results in the Results field.
OK	Saves the calculation and the entered data to the tree view.
Results	The field where the calculated results are displayed (see Figure 13 for an example).

2.3.1. Laminar Calculation – Additional Details Tab

Used to add additional details for the Laminar calculation.

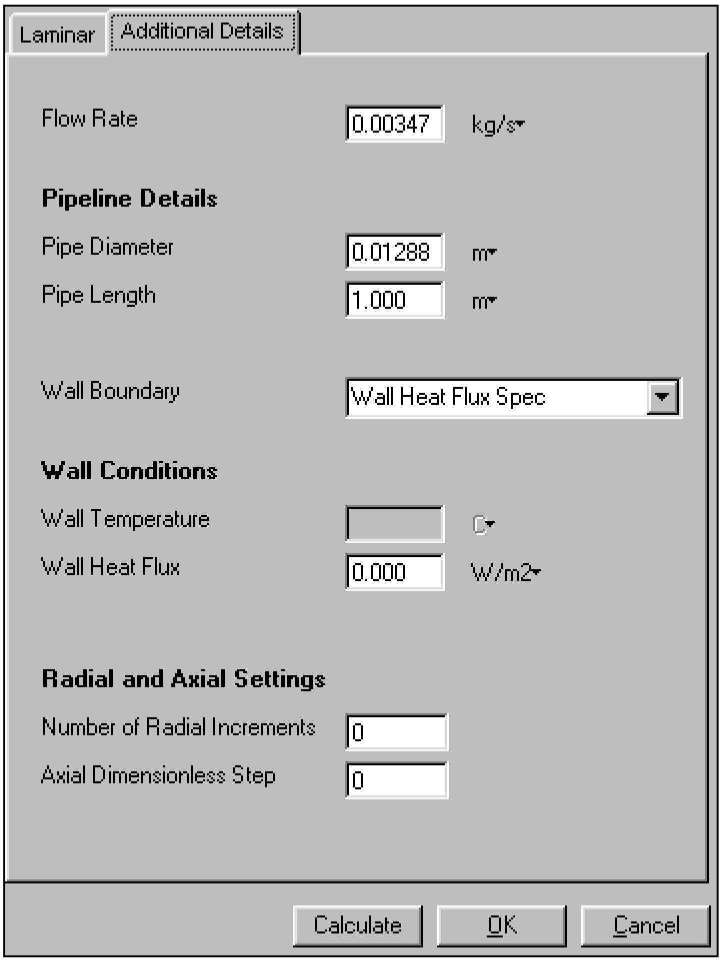


Figure 6 Additional Details dialogue

The following selections can be made from this dialogue:

Usage

Flow Rate	Mandatory. Enter a value for the flow rate down the pipe.
Pipeline Details	Mandatory. Enter values for: • Pipe Diameter • Pipe Length **Wall Boundary** Select from either: • Wall Temperature Spec • Wall Heat Flux Spec **Wall Condition** The following selection can be made when **Wall Temperature Spec** is selected from the drop down menu: • Wall Temperature (greyed out when Wall Heat Flux Spec is selected). The following selection can be made when **Wall Heat Flux Spec** is selected: • Wall Heat Flux (greyed out when Wall Temperature Spec is selected).
Radial and Axial Settings	Enter values for: • Number of Radial Increments • Axial Dimensionless Step.

2.4. Turbulent Calculation

Used to calculate pressure drop and heat transfer for turbulent flow in straight pipes.

Click on the plus sign to expand the tree, the minus sign to collapse it and click on the calculation in the list to display it.

Note. Clicking on a calculation from the tree view more than once duplicates an instance of it and makes it available for comparing results with slight modifications to input.

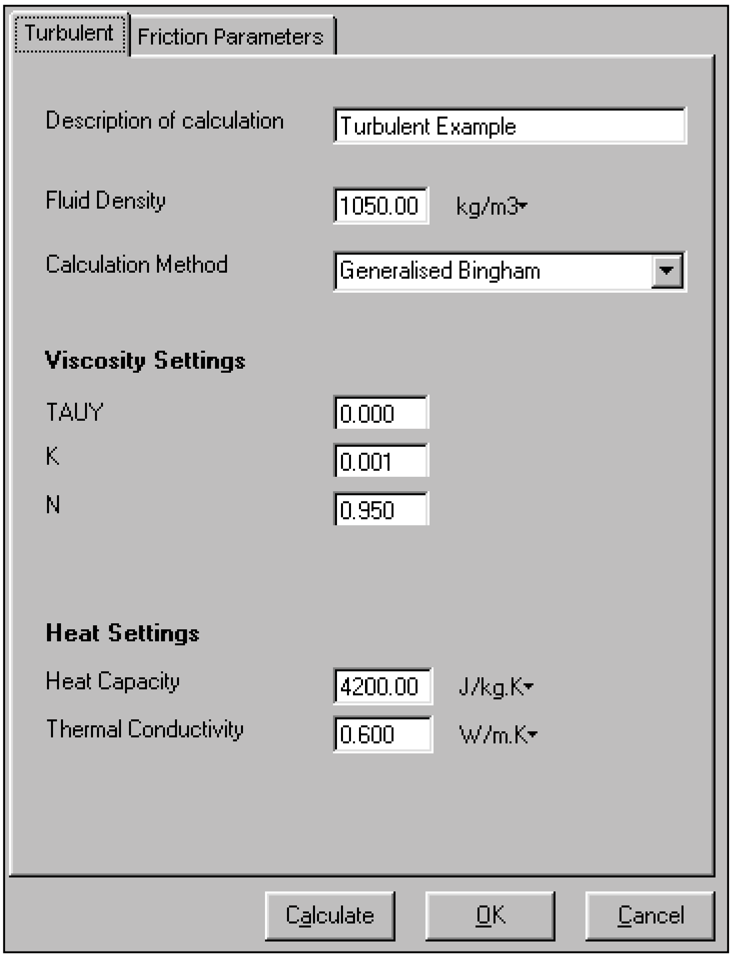


Figure 7 Turbulent Calculation dialogue

The following selections can be made from this dialogue:

Usage

Description of calculation	Mandatory Enter a text string to identify the calculation.
Fluid Density	Mandatory. The actual density of the fluid at operating temperatures.
Calculation Method	Mandatory. Select from either **Newtonian Fluid** or **Generalised Bingham**. The selections available when **Newtonian Fluid** is selected: **Viscosity Settings** • Viscosity at bulk Temperature. **Heat Settings** • Heat Capacity • Thermal Conductivity. The selections available when **Generalised Bingham** is selected: **Viscosity Settings** • TAUY • K • N **Heat Settings** • Heat Capacity • Thermal Conductivity.
Calculate	Performs the calculation and displays any results in the Results field.
OK	Saves the calculation and the entered data to the tree view.
Results	The field where the calculated results are displayed (see Figure 13 for an example).

2.4.1. Turbulent Calculation – Friction Parameters Tab

Used to add additional details for the Laminar calculation.

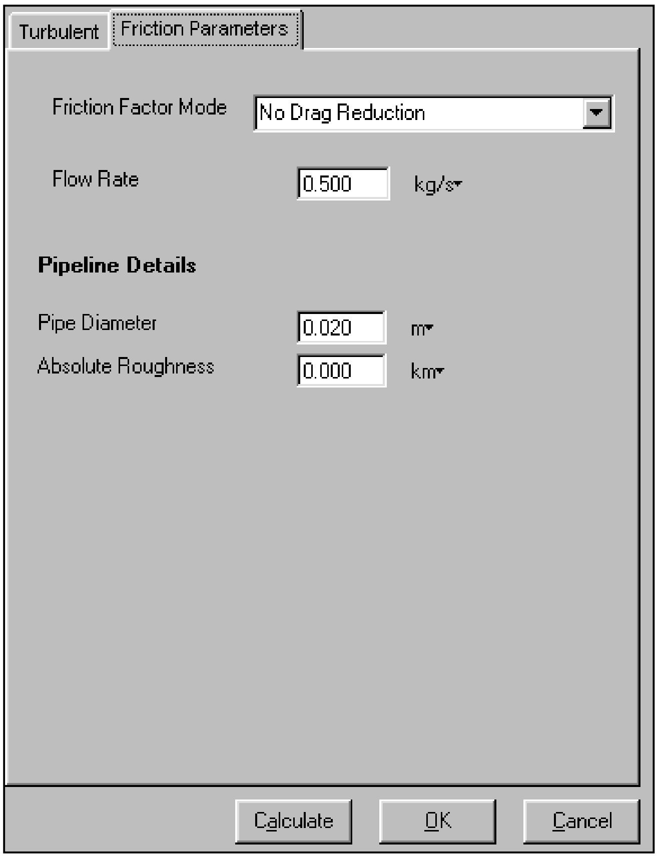


Figure 8 Friction Parameters dialogue

The following selections can be made from this dialogue:

Usage

Friction Factor Mode	Mandatory. Use the drop down menu to select from: **No Drag Reduction** Enter a value for Flow Rate. **Friction Factor versus Reynolds Number** Enter values for: • Flow Rate • Reynolds Number • Friction Factor. Notice that the Reynolds Number and Friction Factor are displayed as a grid. **Friction Velocity versus Degree of Drag Reduction** Enter values for: • Flow Rate • Friction Velocity • Degree of Drag Reduction. Notice that the Friction Velocity and Degree of Drag Reduction are displayed as a grid.
Flow Rate	Mandatory. Enter a value.
Pipeline Details	Mandatory. Enter values for: **Pipe Diameter** Enter the required value or use the Pipe Inner Diameter Calculator (see section 2.5.3.1 for more information). **Absolute Roughness** Enter the required value or use the Pipe Roughness Calculator (see section 2.5.3.2 for more information).

2.5. VisFlo Menus

This section lists the various VisFlo menus and describes the options that are available.

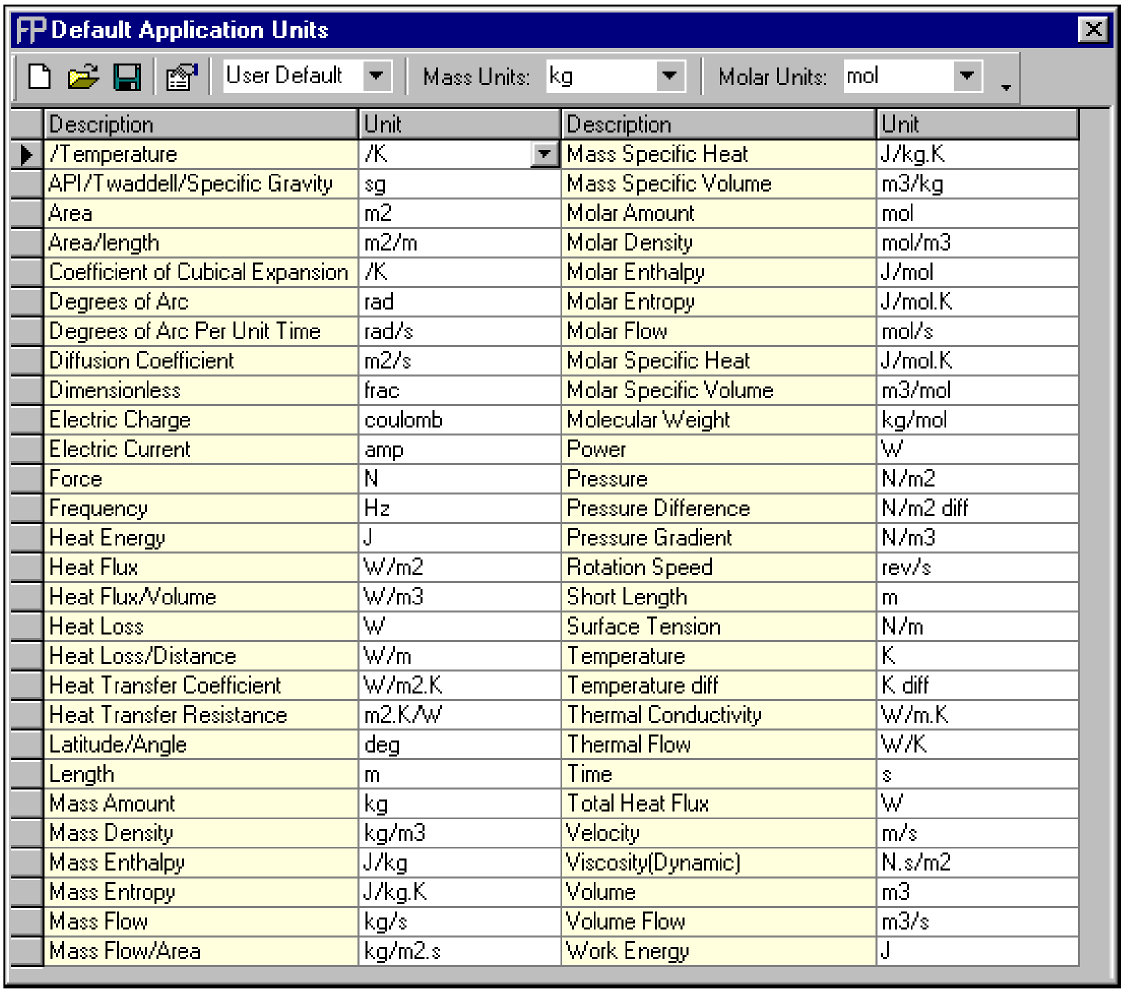
2.5.1. File Menu

The File menu allows the following options to be accessed:

Menu Option	Definition
New	Creates a new Isothermal, Laminar or Turbulent calculation (a maximum of four of each type can be created in one session).
Open	Opens a previously saved calculation.
Save	Saves the active calculation to file.
Print	Prints the active calculated results.
Print Setup	Defines the printer and properties required.
Print Preview	Previews the results file in a text editor.
Exit	Exits the VisFlo program.

2.5.2. Options Menu

The Options menu only allows the Default Units option to be accessed. This option changes the default units used for the majority of input fields.

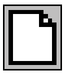


Description	Unit	Description	Unit
/Temperature	/K	Mass Specific Heat	J/kg.K
API/Twaddell/Specific Gravity	sg	Mass Specific Volume	m3/kg
Area	m2	Molar Amount	mol
Area/length	m2/m	Molar Density	mol/m3
Coefficient of Cubical Expansion	/K	Molar Enthalpy	J/mol
Degrees of Arc	rad	Molar Entropy	J/mol.K
Degrees of Arc Per Unit Time	rad/s	Molar Flow	mol/s
Diffusion Coefficient	m2/s	Molar Specific Heat	J/mol.K
Dimensionless	frac	Molar Specific Volume	m3/mol
Electric Charge	coulomb	Molecular Weight	kg/mol
Electric Current	amp	Power	W
Force	N	Pressure	N/m2
Frequency	Hz	Pressure Difference	N/m2 diff
Heat Energy	J	Pressure Gradient	N/m3
Heat Flux	W/m2	Rotation Speed	rev/s
Heat Flux/Volume	W/m3	Short Length	m
Heat Loss	W	Surface Tension	N/m
Heat Loss/Distance	W/m	Temperature	K
Heat Transfer Coefficient	W/m2.K	Temperature diff	K diff
Heat Transfer Resistance	m2.K/W	Thermal Conductivity	W/m.K
Latitude/Angle	deg	Thermal Flow	W/K
Length	m	Time	s
Mass Amount	kg	Total Heat Flux	W
Mass Density	kg/m3	Velocity	m/s
Mass Enthalpy	J/kg	Viscosity(Dynamic)	N.s/m2
Mass Entropy	J/kg.K	Volume	m3
Mass Flow	kg/s	Volume Flow	m3/s
Mass Flow/Area	kg/m2.s	Work Energy	J

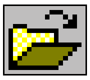
Figure 9 Default Application Units window

The following selections can be made:

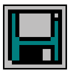
Usage



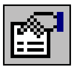
New
Creates a new default units template.



Open
Opens an existing default units template to use.



Save
Saves the current units template under a different file name.



Save as Default Units
Default Units can be selected from:

- Engineering
- British
- SI
- User Default

The units can be changed and saved by performing the following:

1. Click in the unit field and select the appropriate type from the pull down list.
2. Click on the **Save Default Units** icon.

Mass Units Mass Units can be specified as one of the following:
g, kg, tonne, ton, US ton, cwt, lb or slug.

Molar Units Molar Units can be specified as one of the following:
mol, kmol, gmol or lbmol.

▼ Clicking on the down arrow ▼ to the right of the 'Molar Units' field displays an option 'Add or Remove Buttons' that allows the following menu options to be added or removed:

- New
- Open
- Save
- Save as default
- User:
- Mass Units:
- Molar Units:

Resize toolbar and customise facilities are available.

2.5.3. Tools Menu

The Tools menu allows the following options to be accessed:

- Pipe Inner Diameter Calculator
- Pipe Roughness Calculator.

These are described in the following sections:

2.5.3.1. Pipe Inner Diameter Calculator

This dialogue is used to calculate the pipe inner diameter.

Select a standard pipe size then the available schedules for that pipe size. The details for that combination are displayed at the bottom of the dialogue box.

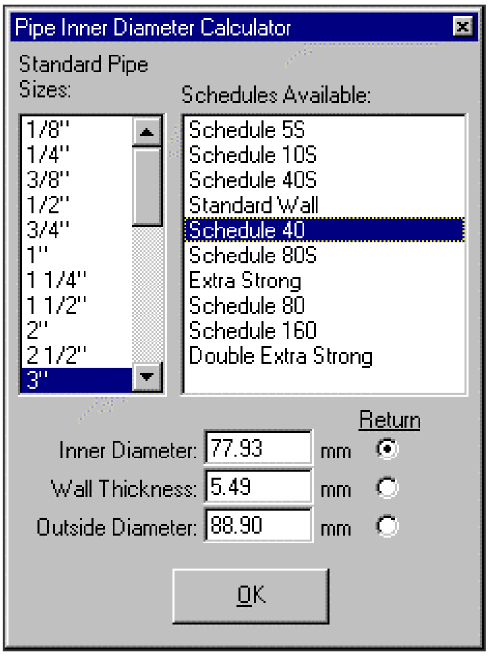


Figure 10 Pipe Inner Diameter Calculator dialogue

The following selections can be made from this dialogue:

Usage

Standard Pipe Sizes	Lists the pipe sizes available from 1/8″ to 36″.
Schedules Available	Lists the pipe schedules available for the pipe size selected.
Inner Diameter	Displays the Inner Diameter of the selected pipe.
Wall Thickness	Displays the Wall Thickness of the selected pipe.
Outside Diameter	Displays the Outside Diameter of the selected pipe.
Return	Use the radio button to indicate which selection is to be returned to the program. The default is Inner Diameter.
OK	Clicking on the OK button returns the chosen value to the program. The value is pasted into the cell that was highlighted when the Pipe Inner Diameter Calculator was selected.

Procedure

1. Click in the field where the pipe inner diameter values are to be placed. Go to the Tools menu and select **Pipe Inner Diameter Calculator**.

2. Select the required pipe size and schedule then click the **Return** radio button to the right of the Inner Diameter box to return the values.

3. Click on **OK**. The values are automatically pasted into the field.

2.5.3.2. Pipe Roughness Calculator

This calculator is used to select a Surface Type and Absolute Roughness. Clicking on OK then adds the selection to the calculation.

The units of measurement for roughness depend on the program calling the Pipe Roughness Calculator. Those for PIPER are in mm. For other programs (for example, FLONET) the Pipe Roughness Calculator displays and returns the relative roughness based on the relevant pipe diameter.

By default, the units are in mm for PEW and the Absolute Roughness is returned.

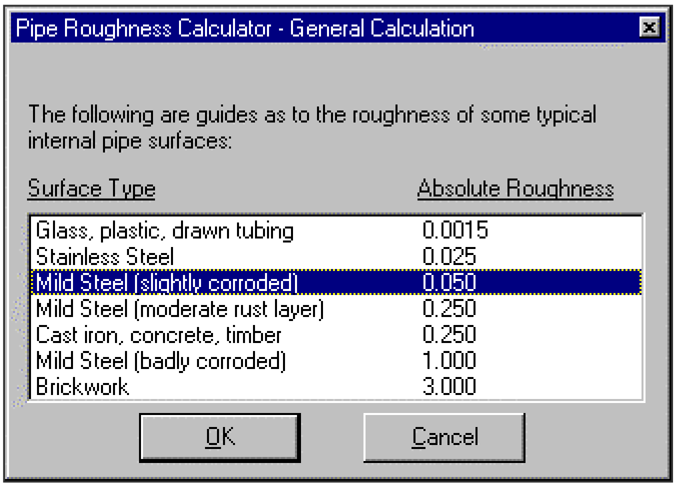


Figure 11 Pipe Roughness Calculator dialogue

Procedure

1. Click in the field where the absolute roughness value is to be placed. Go to the Tools menu and select **Pipe Roughness Calculator**.

2. Select the required surface type.

3. Click on **OK**. The value is automatically pasted into the field.

2.5.4. Window Menu

The Window menu allows the following standard Windows options to be accessed:

- Tile (horizontal and vertical)
- Cascade.

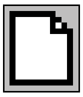
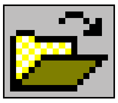
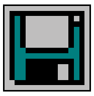
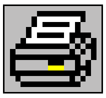
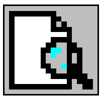
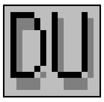
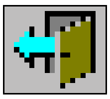
2.5.5. Help Menu

The Help menu allows the following options to be accessed:

- Access the VisFlo help menu for Contents, Index and Search
- Link to the Technical Support topic within the help system
- Access the ABB web site
- Access the PEL Helpdesk web site for reporting problems
- View the About VisFlo dialogue.

2.6. The VisFlo Toolbar

The buttons on the toolbar allow the following options to be selected:

Button	Purpose
	**New Calculation** Creates a new Isothermal, Laminar or Turbulent calculation. Equivalent **File** menu item – New
▼	Clicking on the down arrow ▼ next to the 'New Calculation' button displays the **More Tools** menu that allows the following menu options to be added or removed: • Isothermal • Laminar • Turbulent.
	**Open VisFlo Calculation File** Opens a previously saved calculation. Equivalent **File** menu item – Open
	**Save Calculation Results File** Saves the active calculation to file. Equivalent **File** menu item – Save
	**Print Calculation Results File** Prints the active calculated results. Equivalent **File** menu item – Print
	**View Current Calculation Results** Previews the results file in a text editor. Equivalent File menu item – Print Preview
	**Configure Default Units for Input** Changes the default units used for the majority of input fields. Equivalent **Options** menu item – Default Units
	**Exit VisFlo** Exits the VisFlo program. Equivalent **File** menu item – Exit
▼	Clicking on the down arrow ▼ next to the 'Exit VisFlo' button displays the **More Tools** menu that allows the following menu options to be added or removed: • New • Open • Save • Print • Print Preview • Default Units • Exit. Resize toolbar and customise facilities are available.

3. VisFlo Tutorial

3.1. General

This tutorial contains examples of how to use VisFlo. These are:

Example 1 – Creating an Isothermal Pressure Drop Calculation

Example 2 – Creating a Laminar Flow Heat Transfer Calculation

Example 3 – Creating a Turbulent Flow Heat Transfer Calculation.

3.2. Example 1 – Creating an Isothermal Pressure Drop Calculation

For steady flow of a liquid in a pipe system between sections 1 and 2, a mechanical energy balance can be written as:

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + Z_1 = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + Z_2 + \Sigma h$$

where:

P = absolute pressure
V = velocity
Z = elevation above an arbitrary datum
α = kinetic energy correction factor
Σ h = the sum of the head losses for all straight pipe sections and all fittings.

Either pressure drop or flowrate can be specified and the other calculated. Details of the calculation procedures and correlations used are given in Chapter 3 of the non-Newtonian flow design guide.

Example:

The following example illustrates the use of VisFlo for isothermal pressure drop calculations.

A non-Newtonian liquid is to be transferred under gravity flow from a header tank to a lower tank. The pipeline is 15m long with an inner diameter of 0.0762m.

A schematic representation of the pipe system is shown in Figure 12 below:

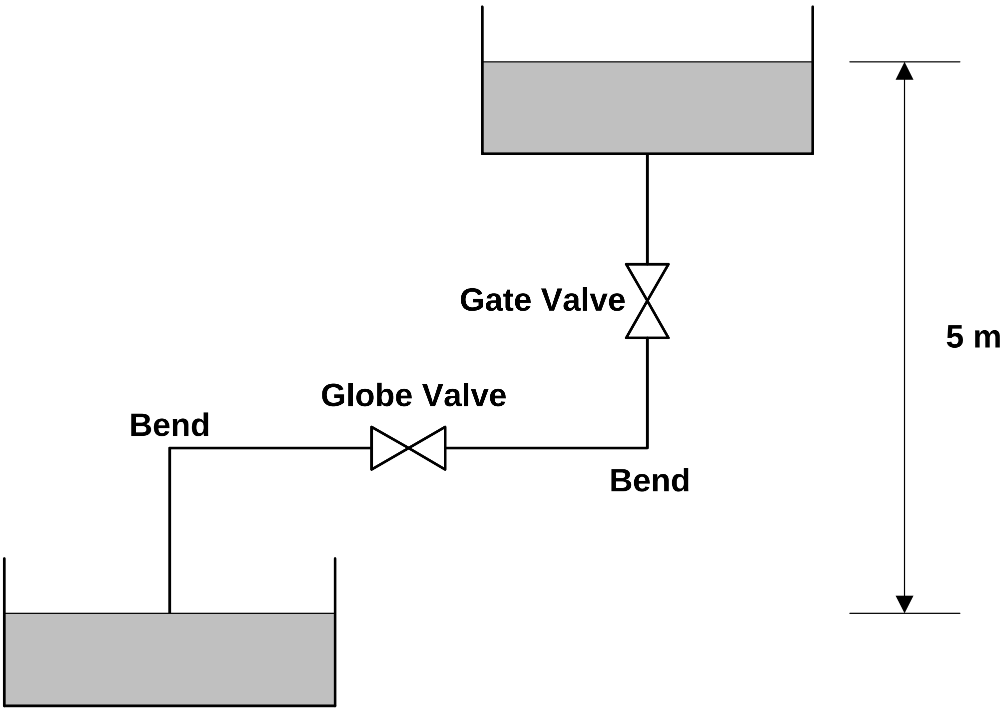
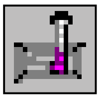


Figure 12 Isothermal Pressure Drop Calculation example

3.2.1. Accessing VisFlo

Procedure



1. Access VisFlo by selecting **Start | Programs | PEL |** then click on the **VisFlo** icon (see left).

2. A splash screen showing the program name and version number appears briefly before the start up screen (see Figure 1) is displayed.

3.2.2. Adding the Isothermal data

Procedure

1. Go to the File menu and select **New** then ▸ **Isothermal**. The Isothermal Calculation window is displayed.

2. Enter a text string in the **Description of calculation** field to identify the calculation.

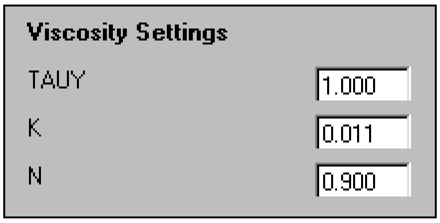
3. Go to the **Fluid Density** field and enter **1000 kg/m^3**.

4. Go to the **Calculation Method** field and select **Generalised Bingham** from the drop down list.

5. Go to **Viscosity Settings** and enter the following values:

   TAUY: **1.0**
   K: **0.011**
   N: **0.9**

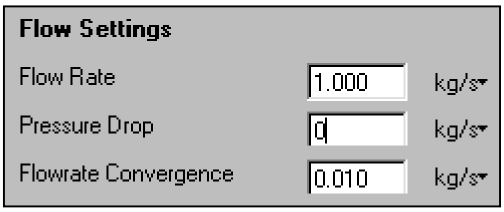
6. The Viscosity Settings fields now look like this:



7. Go to the Calculation Mode field. Select **Pressure Drop Fixed** from the drop down list and enter the following values in the Flow Settings fields:

   Flow Rate: **1.0 kg/s**
   Pressure Drop: **0.0 N/m2**
   Flowrate Convergence: **0.01 kg/s**

8. The Flow Settings fields now look like this:



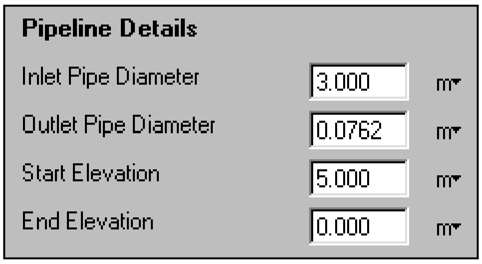
3.2.3. Adding the Pipeline Details data

Procedure

1. Click on the **Pipeline Details** tab and enter the following values in the fields displayed:

Inlet Pipe Diameter:	**3.0 m**
Outlet Pipe Diameter:	**0.0762 m**
Start Elevation:	**5.0 m**
End Elevation:	**0.0 m**

2. The Pipeline Details data fields now look like this:



3.2.4. Adding the Conduits data

Procedure

1. Click on the **Conduits** tab and enter the following values in the fields displayed:

2. Go to the ‘Conduit Type’ drop down list and select **Sudden Contraction**. Enter the following values:

Upstream Diameter:	**3.0 m**
Downstream Diameter:	**0.0762 m**
Number Off:	**5.0 m**

3. Click on the **Add** button. Sudden Contract appears in the left-hand panel.

4. Go to the ‘Conduit Type’ drop down list and select **Pipe Straight**. Enter the following values:

Diameter:	**0.0762 m**
Absolute Roughness:	**0.0 m**
Length:	**15.0 m**
Number Off:	**1**

5. Click on the **Add** button. Pipe Straight appears in the left-hand panel.

6. Select **Gate Valve** from the Conduit Type drop down list and enter the following values:

Diameter:	**0.0762 m**
Number Off:	**1**

7. Click on the **Add** button. Gate Valve appears in the left-hand panel.

8. Select **Elbow** from the Conduit Type drop down list and enter the following values:

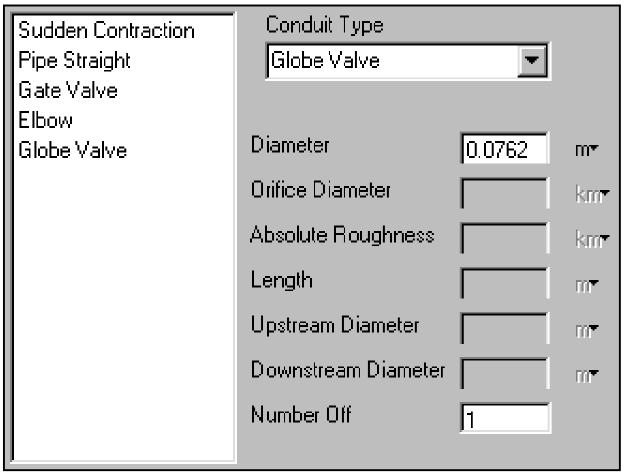
   Diameter: **0.0762 m**
   Number Off: **2**

9. Click on the **Add** button. Elbow appears in the left-hand panel.

10. Select **Globe Valve** from the Conduit Type drop down list and enter the following values:

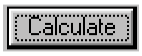
    Diameter: **0.0762 m**
    Number Off: **1**

11. The field to the left of the Conduit Type selector now looks like this:



3.2.5. Performing the calculation

Procedure



1. Click on the **Calculate** button (see left). The results of the calculation are displayed in the Results field on the right-hand side of the screen as shown below:

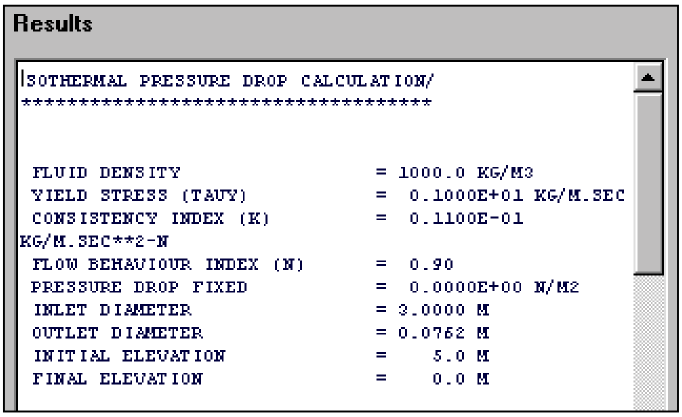
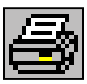


Figure 13 Isothermal Calculation Results (part of)



2. A paper copy of the results can be printed either by clicking on the Print toolbar button (see left) or go to the File menu and select Print. The complete results file is printed in the format shown on the following page.

Note. The output units shown in the results are predetermined and cannot be changed.

```
ISOTHERMAL PRESSURE DROP CALCULATION/ ********************************

 FLUID DENSITY                 = 1000.0 KG/M3
 YIELD STRESS (TAUY)           =  0.1000E+01 KG/M.SEC
 CONSISTENCY INDEX (K)         =  0.1100E-01 KG/M.SEC**2-N
 FLOW BEHAVIOUR INDEX (N)      =  0.90
 PRESSURE DROP FIXED           =  0.0000E+00 N/M2
 INLET DIAMETER                = 3.0000 M
 OUTLET DIAMETER               = 0.0762 M
 INITIAL ELEVATION             =    5.0 M
 FINAL ELEVATION               =    0.0 M

 CONDUIT NUMBER  1
 -----------------
 SUDDEN CONTRACTION
 UPSTREAM DIAMETER             = 3.0000 M
 DOWNSTREAM DIAMETER           = 0.0762 M
 NUMBER OF ITEMS OF THIS TYPE  =  1

 CONDUIT NUMBER  2
 -----------------
 STRAIGHT PIPE
 DIAMETER                      = 0.0762 M
 ROUGHNESS                     =  0.0000E+00 M
 LENGTH                        =   15.0 M
 NUMBER OF ITEMS OF THIS TYPE  =  1

 CONDUIT NUMBER  3
 -----------------
 GATE VALVE
 DIAMETER                      = 0.0762 M
 NUMBER OF ITEMS OF THIS TYPE  =  1



 CONDUIT NUMBER  4
 -----------------
 ELBOW
 DIAMETER                      = 0.0762 M
 NUMBER OF ITEMS OF THIS TYPE  =  2

 CONDUIT NUMBER  5
 -----------------
 GLOBE VALVE
 DIAMETER                      = 0.0762 M
 NUMBER OF ITEMS OF THIS TYPE  =  1
1CONDUIT HEAD LOSSES
 -------------------

 -----------------------------------------------------------------
 : CONDUIT : CONDUIT : VELOCITY : REYNOLDS :NUMBER OF :   HEAD    :
 : NUMBER  :  TYPE   : (M/SEC)  :  NUMBER  : VELOCITY :   LOST    :
 :         :         :          :          :  HEADS   :   (M)     :
 :         :         :          :          :   LOST   :           :
 -----------------------------------------------------------------
 :    1    :    2    :   1.65   : 1.98E+04 :   0.72   :   0.10    :
 :    2    :    1    :   1.65   : 1.98E+04 :   5.04   :   0.70    :
 :    3    :    6    :   1.65   : 1.98E+04 :   2.10   :   0.29    :
 :    4    :    5    :   1.65   : 1.98E+04 :   0.90   :   0.12    :
 :    5    :    7    :   1.65   : 1.98E+04 :  25.40   :   3.52    :
 -----------------------------------------------------------------

 -----------------------------------------------------------------
 : CONDUIT : CONDUIT : VELOCITY : REYNOLDS :   WALL   : APPARENT :
 : NUMBER  :  TYPE   : (M/SEC)  :  NUMBER  :   SHEAR  : VISCOSITY:
 :         :         :          :          :   RATE   : (N.S/M2) :
 :         :         :          :          :   (1/S)  :          :
 -----------------------------------------------------------------
 :    1    :    2    :   1.65   : 1.98E+04 : 1.11E+03 : 7.85E-03 :
 :    2    :    1    :   1.65   : 1.98E+04 : 1.11E+03 : 7.85E-03 :
 :    3    :    6    :   1.65   : 1.98E+04 : 1.11E+03 : 7.85E-03 :
 :    4    :    5    :   1.65   : 1.98E+04 : 1.11E+03 : 7.85E-03 :
 :    5    :    7    :   1.65   : 1.98E+04 : 1.11E+03 : 7.85E-03 :
 -----------------------------------------------------------------
 CALCULATED FLOWRATE           =   7.52 KG/SEC
 CALCULATED PRESSURE DROP      =  1.2686E+01 N/M2
```

Figure 14 Isothermal Calculation Results

3.3. Example 2 – Creating a Laminar Flow Heat Transfer Calculation

This procedure describes the steps to generate a Laminar Flow Heat Calculation.

Details of the calculation procedure and correlations used in the laminar flow heat transfer calculation are given in Chapter 6 of the non-Newtonian flow design guide. The calculation takes account of the variation of viscous properties with temperature and internal heat generation from viscous dissipation.

Two alternative boundary conditions can be specified:

1. Constant wall temperature

2. Constant wall heat flux (a positive value indicates heat input).

Example:

A molten polymer at an initial temperature of 200 degrees C is to be pumped through a straight pipe 1m long with an Inner Diameter of 0.01288m. There is to be no heat flux through the pipe wall.

3.3.1. Adding the Laminar Heat Transfer data

Procedure

1. Access VisFlo as described in section 3.2.1.

2. Go to the File menu and select **New** then ▸ **Laminar**. The Laminar Calculation window is displayed.

3. Enter a text string in the **Description of calculation** field to identify the calculation.

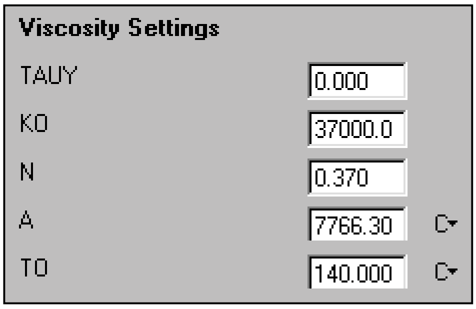
4. Go to the **Fluid Density** field and enter **766.0 kg/m^3**.

5. Go to the **Calculation Method** field and select **Generalised Bingham** from the drop down list.

6. Go to **Viscosity Settings** and enter the following values:

TAUY:	**0.0**
KO:	**0.011**
N:	**37000.0**
A:	**7766.30**
N:	**140.0**

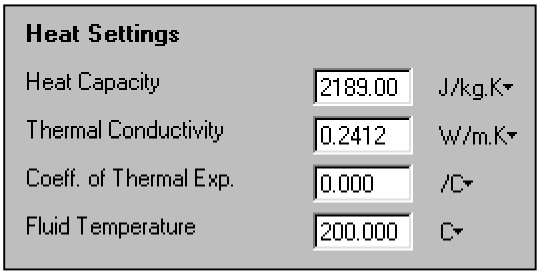
9. The Viscosity Settings fields now look like this:



10. Go to **Heat Settings** and enter the following values:

Heat Capacity:	**2189.00**
Thermal Conductivity:	**0.2412**
Coefficient of Thermal Expansion:	**0.0**
Fluid Temperature:	**200.0**

11. The Heat Settings fields now look like this:



3.3.2. Adding Additional Details

Procedure

1. Go to the **Flow Rate** field and enter **000347 kg/s**.

2. Go to **Pipeline Details** and enter the following values:

Pipe Diameter:	**0.01288 m**
Pipe Length:	**1.0 m**

3. Go to the **Wall Boundary** field and select **Wall Heat Flux Spec** from the drop down list.

4. Go to **Wall Conditions** and enter the following value:

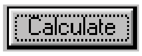
Wall Heat Flux: **0.0**

5. Go to **Radial and Axial Settings** and enter the following values:

Number of Radial Increments:	**0.0**
Axial Dimensionless Step:	**0.0**

3.3.3. Performing the calculation

Procedure



1. Click on the **Calculate** button (see left). The results of the calculation are displayed in the Results field on the right-hand side of the screen as shown below:

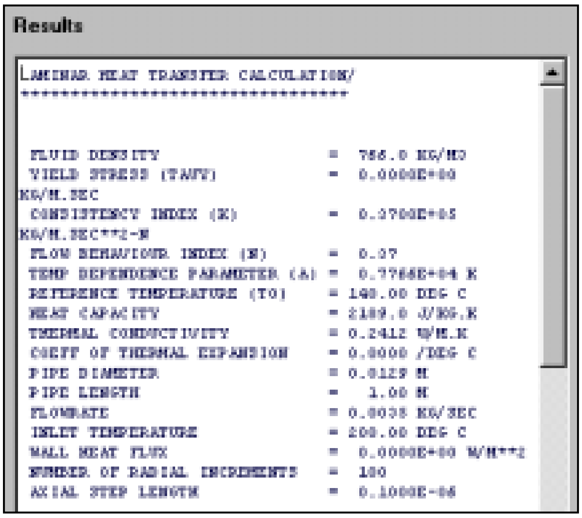
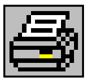


Figure 15 Laminar Calculation Results (part of)



2. A paper copy of the results can be printed either by clicking on the Print toolbar button (see left) or go to the File menu and select Print.

   The complete results file is printed in the format shown on the following page.

Note. The output units shown in the results are predetermined and cannot be changed.

```
LAMINAR HEAT TRANSFER CALCULATION/ *********************************


 FLUID DENSITY                  =  766.0 KG/M3
 YIELD STRESS (TAUY)            =  0.0000E+00 KG/M.SEC
 CONSISTENCY INDEX (K)          =  0.3700E+05 KG/M.SEC**2-N
 FLOW BEHAVIOUR INDEX (N)       =  0.37
 TEMP DEPENDENCE PARAMETER (A)  =  0.7766E+04 K
 REFERENCE TEMPERATURE (TO)     = 140.00 DEG C
 HEAT CAPACITY                  = 2189.0 J/KG.K
 THERMAL CONDUCTIVITY           = 0.2412 W/M.K
 COEFF OF THERMAL EXPANSION     = 0.0000 /DEG C
 PIPE DIAMETER                  = 0.0129 M
 PIPE LENGTH                    =   1.00 M
 FLOWRATE                       = 0.0035 KG/SEC
 INLET TEMPERATURE              = 200.00 DEG C
 WALL HEAT FLUX                 =  0.0000E+00 W/M**2
 NUMBER OF RADIAL INCREMENTS    =  100
 AXIAL STEP LENGTH              =  0.1000E-06


 FLUID VELOCITY                 =  0.035 M/SEC
 REYNOLDS NUMBER                =  0.2240E-04
 PRESSURE DROP                  =  0.1517E+08 N/M**2


 FINAL VELOCITY PROFILE ( CENTRE TO WALL IN M/SEC ) :

   0.052    0.052    0.052    0.052    0.051    0.051    0.051    0.051    0.051
   0.051    0.051    0.051    0.051    0.051    0.051    0.051    0.051    0.050
   0.050    0.050    0.050    0.050    0.049    0.049    0.049    0.048    0.048
   0.047    0.047    0.046    0.045    0.044    0.043    0.042    0.041    0.040
   0.039    0.037    0.035    0.034    0.032    0.029    0.027    0.024    0.022
   0.019    0.015    0.012    0.008    0.004    0.000


 NUSSELT NUMBER                 =     0.00
 HEAT TRANSFER COEFFICIENT      =     0.00 W/M**2 K
 FINAL BULK TEMPERATURE         =   209.14 DEG C


 FINAL TEMPERATURE PROFILE ( CENTRE TO WALL IN DEG C ) :

   201.5    201.6    201.6    201.6    201.6    201.7    201.8    201.9    202.0
   202.1    202.2    202.4    202.5    202.7    202.9    203.1    203.3    203.6
   203.9    204.2    204.5    204.8    205.2    205.6    206.0    206.4    206.9
   207.3    207.8    208.4    208.9    209.4    210.0    210.6    211.2    211.8
   212.4    213.0    213.6    214.2    214.8    215.4    215.9    216.4    216.9
   217.3    217.7    218.0    218.2    218.3    218.4
```

Figure 16 Laminar Calculation Results

3.4. Example 3 – Creating a Turbulent Flow Heat Transfer Calculation

Details of the calculation procedure and correlations used in the turbulent flow heat transfer calculation are given in Chapter 7 of the non-Newtonian flow design guide. Fluid properties at the average bulk temperature should be used.

Friction factors can be determined by one of three different methods:

1. Colebrook-White equation for friction factor versus Reynolds number
2. User supplied friction factor versus Reynolds number data
3. User supplied degree of drag reduction versus friction velocity data.

Reynolds numbers are based on the apparent viscosity at the wall. If user supplied data are to be used, a maximum of 20 data points can be entered and linear interpolation is carried out between data points.

This procedure describes the steps to generate a Laminar Flow Heat calculation.

The following example illustrates the use of VisFlo to determine.

Example:

A slurry is known to be not subject to drag reduction and is to be heated on the tube side of a shell and tube heat exchanger. The tube Inner Diameter is 0.02 m and the Flowrate in each tube is 0.5 kg/sec.

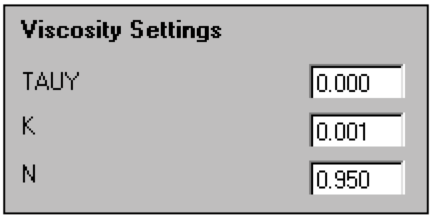
3.4.1. Adding the Turbulent Heat Transfer data

Procedure

1. Access VisFlo as described in section 3.2.1.
2. Go to the File menu and select **New** then ▸ **Turbulent**. The Turbulent window is displayed.
3. Enter a text string in the **Description of calculation** field to identify the calculation.
4. Go to the **Fluid Density** field and enter **1050.0 kg/m^3**.
5. Go to the **Calculation Method** field and select **Generalised Bingham** from the drop down list.
6. Go to **Viscosity Settings** and enter the following values:

   TAUY: **0.0**
   KO: **0.001**
   N: **0.95**

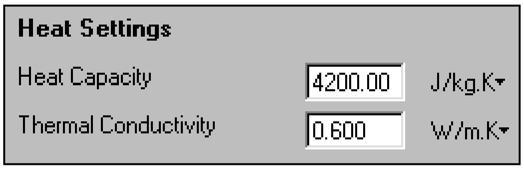
7. The Viscosity Settings fields now look like this:



8. Go to **Heat Settings** and enter the following values:

Heat Capacity: **4200.00**
Thermal Conductivity: **0.6**

9. The Heat Settings fields now look like this:



3.4.2. Adding Friction Parameters data

Procedure

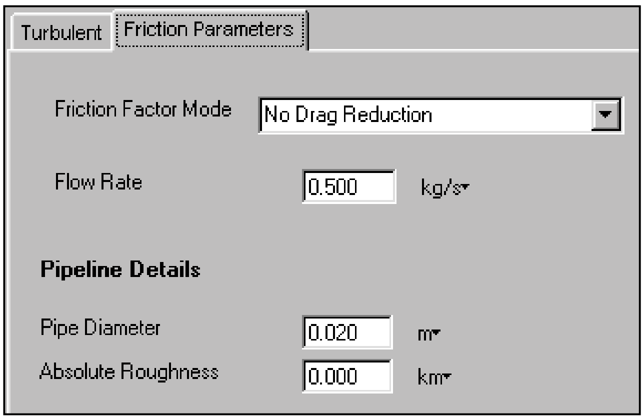
1. Go to the **Friction Factor Mode** field and select **No Drag Reduction** from the drop down list.

2. Go to the **Flow Rate** field and enter **0.500 kg/s**.

3. Go to **Pipeline Details** and enter the following values:

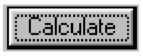
Pipe Diameter: **0.02 m**
Pipe Length: **0.0 m**

4. The Friction Parameters dialog now looks like this:



3.4.3. Performing the calculation

Procedure



1. Click on the **Calculate** button (see left). The results of the calculation are displayed in the Results field on the right-hand side of the screen as shown below:

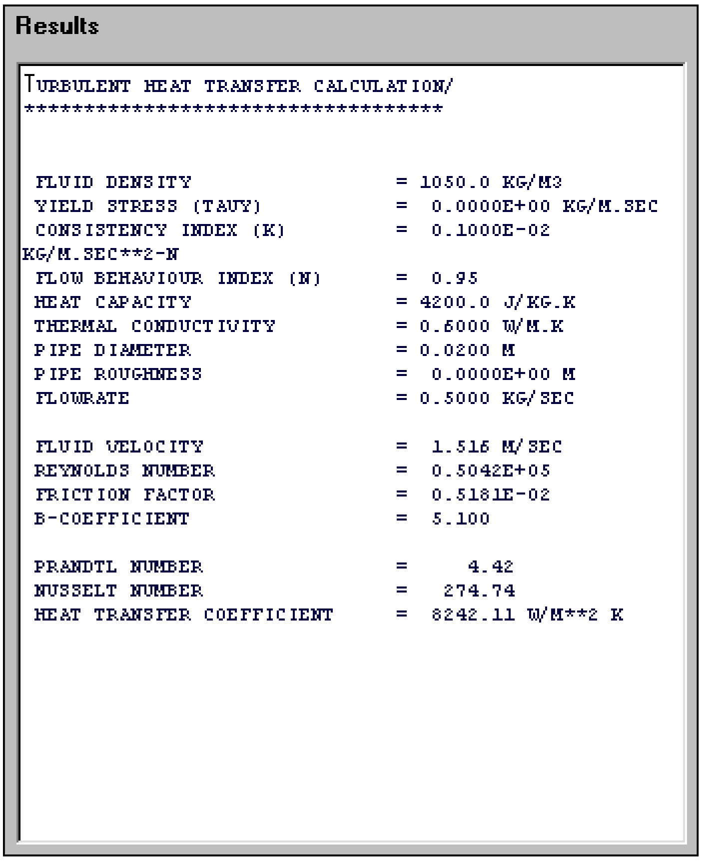
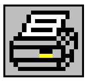


Figure 17 Turbulent Calculation Results (part of)



2. A paper copy of the results can be printed either by clicking on the Print toolbar button (see left) or go to the File menu and select Print.

   The complete results file is printed in the format shown on the following page:

Note. The output units shown in the results are predetermined and cannot be changed.

```
TURBULENT HEAT TRANSFER CALCULATION
***********************************


FLUID DENSITY                  = 1050.0 KG/M3
YIELD STRESS (TAUY)            =  0.0000E+00 KG/M.SEC
CONSISTENCY INDEX (K)          =  0.1000E-02 KG/M.SEC**2-N
FLOW BEHAVIOUR INDEX (N)       =  0.95
HEAT CAPACITY                  = 4200.0 J/KG.K
THERMAL CONDUCTIVITY           =  .6000 W/M.K
PIPE DIAMETER                  =  .0200 M
PIPE ROUGHNESS                 =  0.0000E+00 M
FLOWRATE                       =  .5000 KG/SEC

FLUID VELOCITY                 =  1.516 M/SEC
REYNOLDS NUMBER                =  0.5042E+05
FRICTION FACTOR                =  0.5181E-02
B-COEFFICIENT                  =  5.100

PRANDTL NUMBER                 =     4.42
NUSSELT NUMBER                 =   274.74
HEAT TRANSFER COEFFICIENT      =  8242.11 W/M**2 K
```

Figure 18 Turbulent Calculation Results

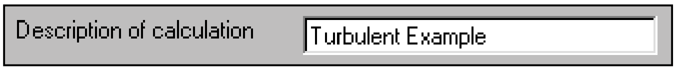
Appendices

Appendix A – Field Input

There are two standard field entries – a text box field and a unit field box. Both of these allow the user to click into or tab to then and enter data.

The Text Box

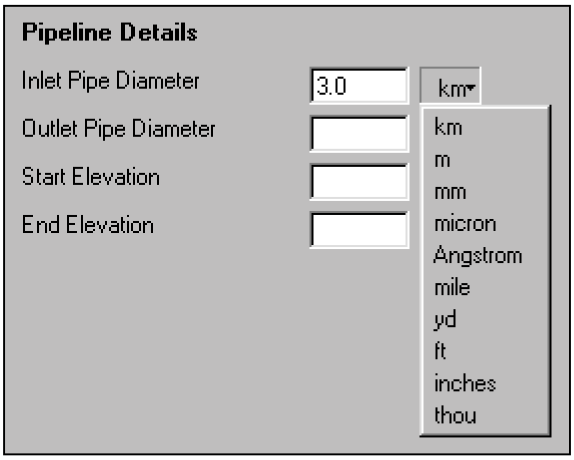
The text box is typical of each calculation and is used to enter a textual description of the type of calculation.



The next text box is set up to allow numerical data such as 1,2,40, etc.

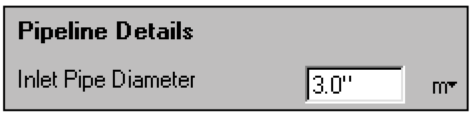
The Unit Field Box

The unit field box comprises of an input box and a unit pull down list. The entered value is dependant on the unit shown.

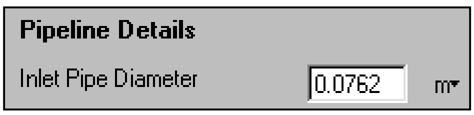


Enter a value followed by a unit parameter which is converted automatically to the displayed unit either when the return key is pressed or the cursor is moved to another field.

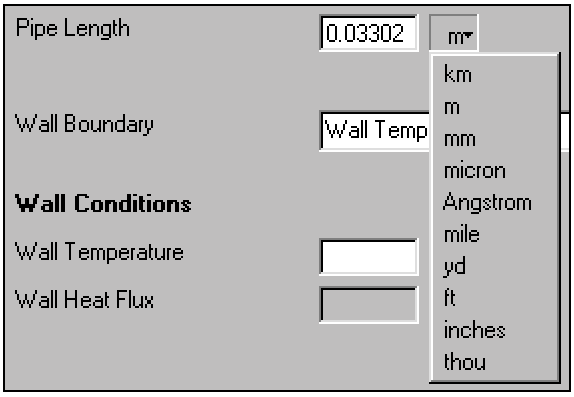
In the following example 3.0 inches is entered and becomes 0.0762 m.



becomes:



The displayed unit can be changed by clicking on it and selecting from the displayed list (see example below). This has the effect of automatically changing any value entered in the field to the new unit type.



Notes

Use these pages to record any notes.

Notes